

CONCEPT OF FINITE ELEMENT MODELLING FOR TRUSSES AND BEAMS USING ABAQUS

Praveen Padagannavar

School of Aerospace, Mechanical & Manufacturing Engineering
Royal Melbourne Institute of Technology (RMIT University)
Melbourne, VIC 3001, Australia

ABSTRACT

Abaqus is one of the powerful engineering software programs which are based on the finite element method. The Abaqus can solve wide range of problems from linear to nonlinear analyses. Abaqus is widely used in many sectors like automotive and mechanical industries for design and development of FEM products. The finite element method is a numerical technique for finding approximate solutions for differential and integral equations. The finite element word was coined by Clough in 1960. In 1960s, engineers used the method for solving the problems in stress analysis, strain analysis, heat and fluid transfer, and other region. Abaqus CAE can provide a simple creating model, submitting the modal, monitoring, and evaluating result and then can also compare with theoretical calculation. In this report all the steps will be explained in different areas such as sketch a modal, assigning material property, applying boundary condition and loads, submit and monitor the job and view the deformed models using the visualisation and create report for results. This report is to demonstrate to create and analyse a structure model in two dimensional with the aim of showing Abaqus software. The methods used to test the modal and analyse on different boundary conditions and also analyse the behaviour of modal with different elements such as truss and beam elements and assume frictionless pin joint in truss and rigid joints welded in beam elements. The results will be compared and explained with theoretical calculated statically determinate truss.

Key words: Abaqus, trusses, beam, simulation and finite element analysis.

Cite this Article Praveen Padagannavar, Concept of Finite Element Modelling for Trusses and Beams using Abaqus. *International Journal of Mechanical Engineering and Technology*, 7(3), 2016, pp. 38–52.

<http://www.iaeme.com/currentissue.asp?JType=IJMET&VType=7&IType=3>

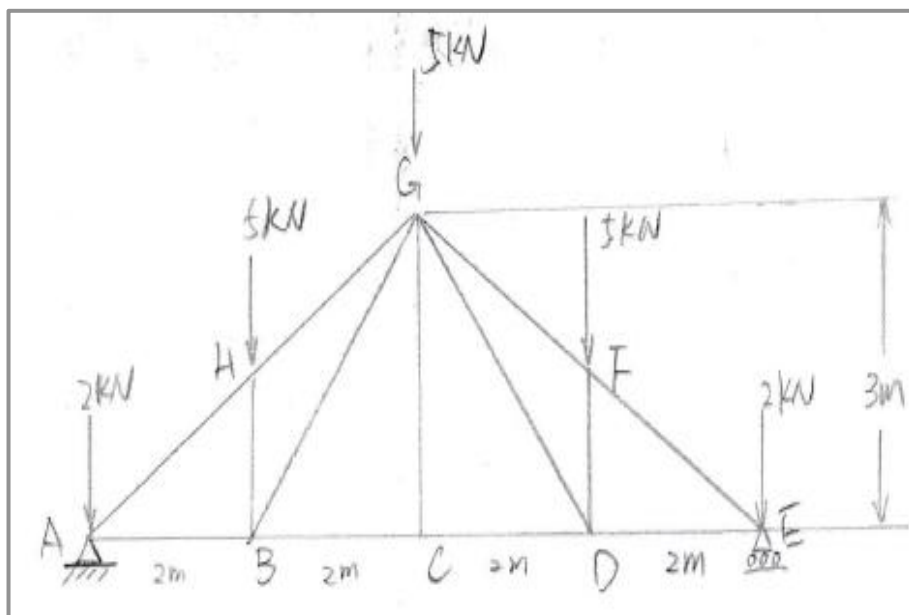
1. INTRODUCTION

Nowadays, the trend is towards new technology and complex advanced structures. The highly structured quality has become a major effort to refine the programs. The aim of this report is to study the structure behaviour with truss and beam elements by using the ABAQUS/CAE software and compare with theoretical of the statically determinate. The Finite Element Analysis is common methods used to analyse static and dynamic, numerical method for solving engineering problems by mathematical. One of the purposes using finite element method is predicting the performance of design, understand the physical behaviours of a modal and identify the weakness of the design accurately to obtain the safety. Two models with different Boundary Conditions and different element type are analysed using Finite Element Method. The numerically solution for the given frames is to yield an approximate solution and for analytical methods which yield an exact solution. The results allow us to analyse the stresses and strains generated in the Frames and predict its deformation. Although, the results are approximate and need to compare with the theoretical results. Theoretical calculation is difficult to solve manually. Finite Element Method is a good option to estimate the response to loads.

The **objective** of this paper is to calculate vertical and horizontal displacements at all nodes, reactions forces and member forces by using finite element analysis and ABAQUS/CAE for given frames and compare with theoretical calculation. This result generated should be close to exact solution and it should have accuracy without being computationally expensive.

2. MODAL DEVELOPMENT

2.1. Hand Sketch



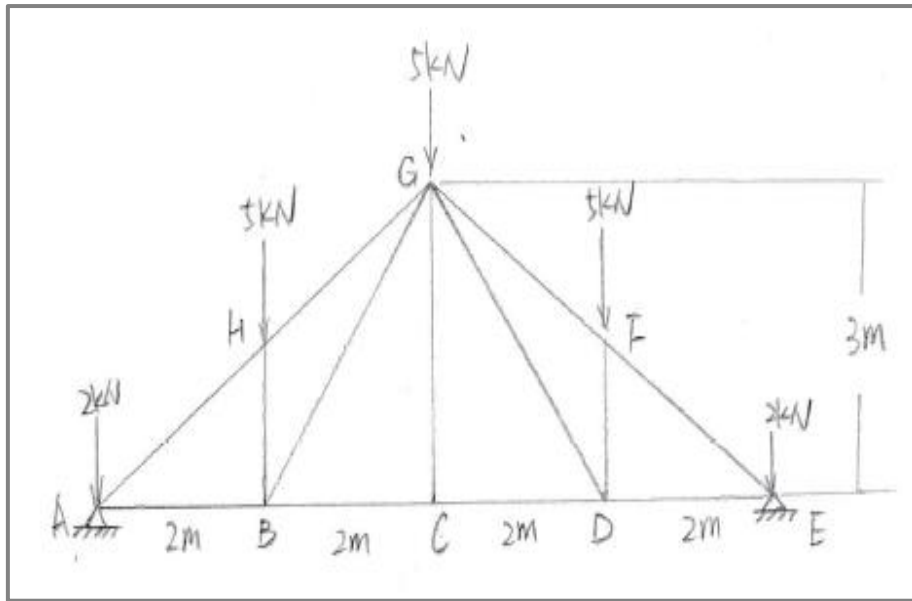


Figure 1. Hand sketch (Fixed support and Roller support)

2.2. Model Geometry Details

At the point H, G, and F the load is 5kN (5000N) and at the point A and E the load is 2 kN (2000N)

Poisson ratio $\nu = 0.3$

$E = 100 \times X$, $X = 1.0 + 0.001 \times 101$

$E = 110.1 \text{e}9 \text{ Pa}$

For truss element: $A = 6400 \text{mm}^2 \rightarrow 0.0064 \text{m}^2$

For beam element: cross section is $80 \text{mm} \times 80 \text{mm} \rightarrow 0.08 \text{m} \times 0.08 \text{m}$

2.3. Steps and Explanations for Truss and Beam

Step 1: Go to program and select Abaqus CAE then the Abaqus window will open select for “with standard modal”.

Step 2: Start with first part “**Module Part**” in this module we need to modal the frame, in this we can create, edit, and manage the part. This is functional units of Abaqus called modules. In our case we are creating modal.

- Click on part and then select part manager.
- In the part manager click on create then the part create new window will open select for 2D planar modelling space, deformable type, wire feature and approximate size and then continue and dismiss the previous window.
- Truss elements can be used two or three dimensions to modal. Two dimensions elements are used for pin joints or bolts.
- In truss we are selecting wire because to connect the two points like rods or connecting two or more points in straight line.
- Create points in the grid coordinates points (x,y) like (0,0),(2,0),(4,0),(-2,0),(-4,0),(0,3).
- Then create the line by selecting the coordinate's points.
- Then at the bottom click on done. Now we created the modal frame.

Step 3: Select the second part that is “**Property Module**” in this module we need to apply material properties to the given modal frame that is define materials, material behaviour and define section. Assign each material property and region of a part.

2.4. In the case of TRUSS:

- Start with **Material** which is located at the top main menu toolbar, click on it and then select on create. Here we are defining material.
- Edit material new window box will open.
- Select on mechanical, change to elasticity – elastic. Linear elastic modal is isotropic and have elastic strain.
- Put the values of Young’s Modulus and Poisson’s Ratio and then click OK. These are parameter area to be defined.
- Secondly, select **Section** in this feature we need to apply cross sectional of the modal frame.
- Create section dialogue box will open then click on **beam—truss** and continue and also put the values of cross sectional area of the modal frame. Here we are selecting beam in truss because trusses are like beam **which is 2 or 3 dimensional rod like structure which has axial but no bending.**
- Finally, select **Assign** and click on section and then select the region to be assigned select entire modal frame and click Done at the bottom. Section properties that have assigned to the part assigned automatically to all instance.

2.5. In the case of BEAM:

- Beam is 2 or 3 dimensional to modal rod like structure that can be axial and bending stiffness. Beam structure has cross sectional area and assigned only to wire region.
- Start with **Material** which is located at the top main menu toolbar, click on it and then select on create. Here we are defining material.
- Edit material new window box will open.
- Select on mechanical, change to elasticity – elastic. Linear elastic modal is isotropic and have elastic strain.
- Put the values of Young’s Modulus and Poisson’s Ratio and then click OK. These are parameter area to be defined.
- Secondly, select **Section** in this feature we need to apply cross sectional of the modal frame.
- Create section dialogue box will open then click on **beam—beam.**
- Edit beam section window will open. Click here to **create beam profile**, select **rectangular profile** and continue. Rectangular profile is geometric data of rectangle solid.
- After continue, another window will open put the values of **rectangular shape a and b** that is 80mmX80mm and click OK. “a” is the length of rectangle parallel of first axis and “b” is the length of rectangle parallel of second axis.
- Finally, select **Assign** and click on section and then select the region to be assigned select entire modal frame and click done at the bottom. Section properties that have assigned to the part assigned automatically to all instance.
- Again select **Assign** and click on **beam section orientation**. Select the entire region to be assigned a beam section orientation and click ok. When you click OK, **tangent vector** are shown (approximate n1 direction) and then press enter to continue and

click OK at the bottom to confirm input. Beam section orientation is assigned to wire region and it defines the orientation is in one direction of the cross section.

Step 3: The third part is “**Assembly Module**”. In our modal we have only one assembly.

- Select **Instance** and click on create to own coordinate system
- In this new window we need to select parts and dependent instance type and click OK. Click only OK, because if we click apply and ok means then we are creating two instances and one is sitting behind the modal, so here is important to click only ok. Dependent is the original part.
- **Step 4:** The fourth part is “**Step Module**”
- Select **step** which is located at the top of the toolbar and click on create. In step we can edit or manipulate the current modal.
- In this new window box change the setting to linear perturbation procedure type and static, linear perturbation and click Continue. Linear perturbation analysis provides linear response of the modal.
- Give description to the step-1 and click Ok.

Step 5: The fifth part is “**Load Module**” in this module we will apply boundary condition and load to the modal frame. Boundary condition fixes the degree of freedom and has two types rotational and translational degree of freedom.

- Select **BC** which is located at the top of the toolbar and click on create. Then create boundary condition dialogue box will open and then change the settings to Initial - mechanical category – displacement/ rotation and then click continue. Select the region to apply BC. Displacement / rotation means holding the movement of selected nodes degree of freedom to 0
- Select the two corner points to of the modal frame.
- For fixed support tick for U1 and U2.
- For roller support tick for U2
- Now it’s time to apply **Load** select for it which is located at the top. We should name the load, type of load and apply.
- Then click on create load, change the setting to Step-1, mechanical and concentrated force (applied to vertices) and click continue. Concentrated force is to the nodes
- Now pick up the points to apply load. In this paper 5kN is applied at the top three points and 2kN is applied at the two end corner points.
- After picking the points when you click done, another window will open this window will show the direction of the load that is CF2. We will use minus sign because load should be applied to opposite direction to the origin.

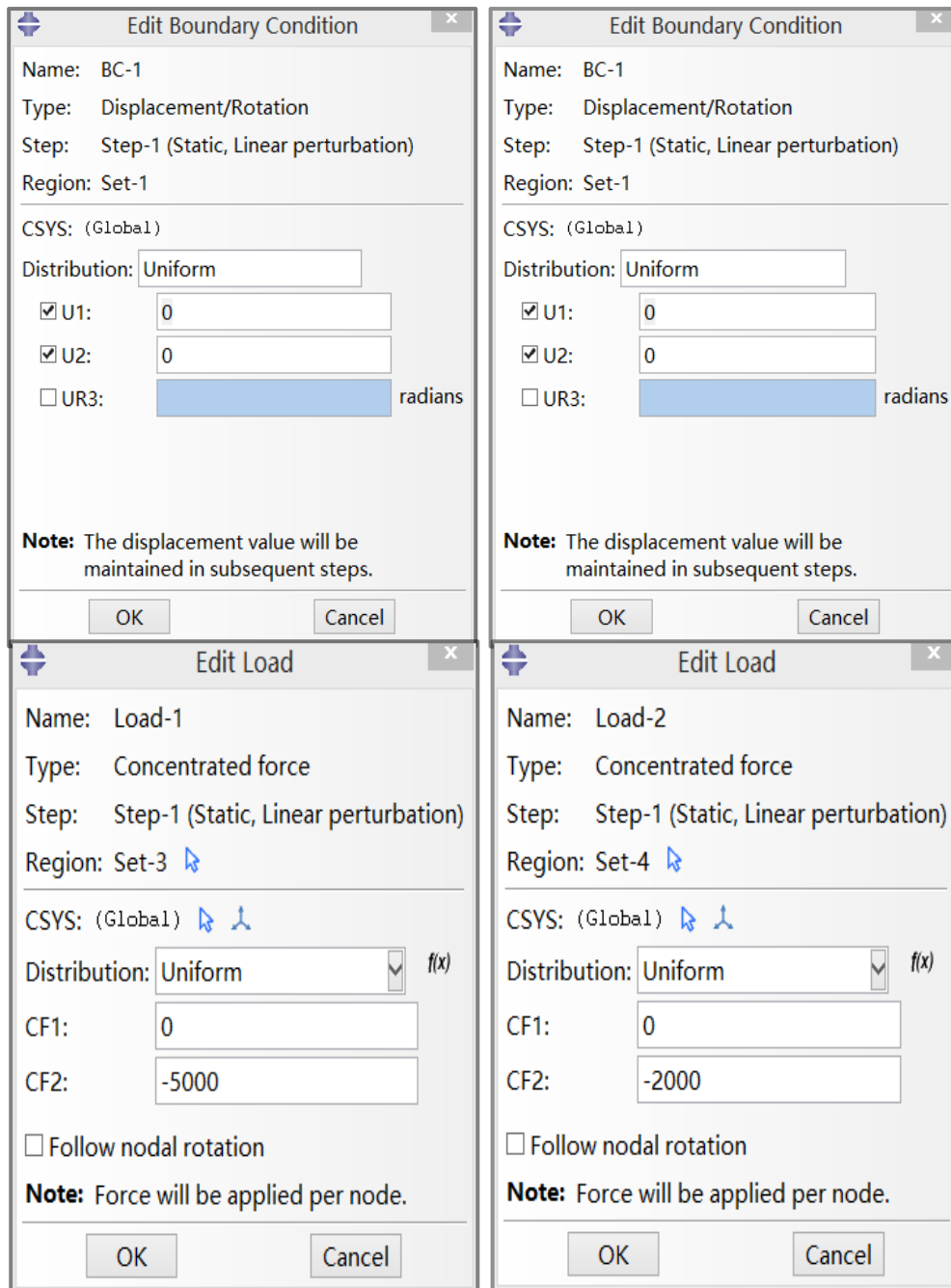


Figure 2: Boundary condition and load applied

Step 6: The sixth part is “**Mesh Module**” in this module we will mesh the modal frame according to the requirement to get proper results. Mesh means converting whole material into small network and also we can define mesh density, mesh shape (1 or 2 or 3 dimensional) and mesh element. The main aim of mesh is to reduce the error while solving the results. We can also mesh by partition so that the mesh structure will be finer and perfect shape. Mesh is created to confirm the node position and element. A higher level of accuracy can be attained by using a fine mesh; however this would be at the cost of more computing power and time.

- Click on Part-1
- First, select **seed** which is located at the top and click on part and put the values of approximate global size seeds and then click OK and Done. Seeding is used to

specify mesh density. Seeds are only located at the edges. While, putting the value we need to select properly otherwise it will show deformation size is large error, that time we must decrease the number. The seeding size chosen is 8.

- Secondly, select **Mesh** and click on **element type**. Select the region to be assigned element type, select the entire modal frame and click Done.
- **In the case of TRUSS:** The new window box will open that is element type, change the settings to **standard-linear-truss** and click OK.
- **In the case of BEAM:** The new window box will open that is element type, change the settings to **standard-linear-Beam** and click OK.
- Finally, again select **Mesh** and click on **part** and then click yes at the bottom mesh the part.

Step 7: The final part is “**Job Module**” in this module we will submit the modal frame for analysis and evaluation and get the results. This is the last step.

- Select the **Job** located at the top and click on create. In this dialogue box name the job and click continue and OK.
- Again select **job** and click on manager and submit the job (modal frame) for evaluation.
- Check for the command “completed successfully”
- Then click on results to view the results.
- Then click on report which is at the top and then click on field output. Give the location to save the **abaqus.rpt**, so that we can check the report.
- Save the modal.
- Results can also be viewed in visualisation module. We can see deformed shape, undeformed shape and contours.
- The report can be generated by using the option field output, unique nodal. Click for stress component, strain components, displacements and reaction forces.

2.6. Boundary Condition

Roller support means fixing and making the model movable only in the x direction and constrained at y-axis.

Fixed support means fixing in the respective x and y direction making the structure rigid. Translational motion in axis 1 and 2 are constrained for both the nodes.

2.7. Mesh

Finite Element Method involves breaking a given structure into smaller element with simple geometry and theoretical solution. The elements are joined to each other **at Nodes**, this procedure is called **Meshing**. The mesh size is important feature in ABAQUS CAE and to get the better results. Finer the coarse mesh sizes of each element better the results. It is important to mesh the model for uneven shapes because at corner of complex model the mesh is irregular, to overcome this partition feature will help to make regular mesh. More the mesh then more accurate results but also requires more time. 20 percent of the time goes to generate the mesh

3. ABAQUS RESULTS

3.1. Truss Element: Fixed and Roller Support

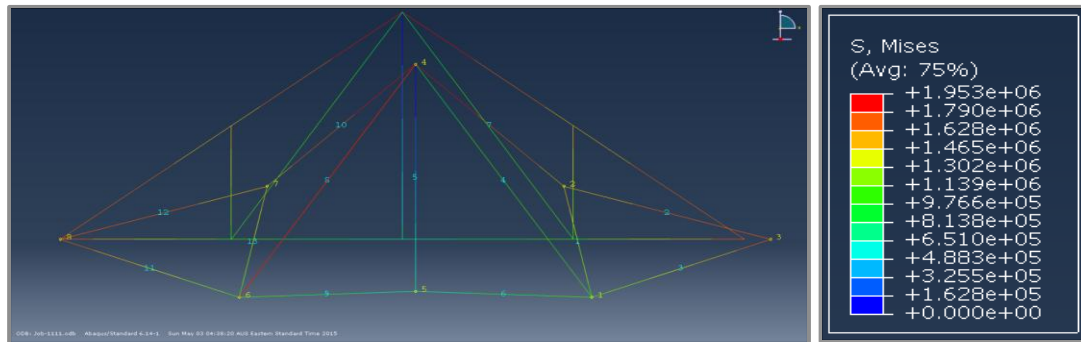


Figure 3: Deformed and undeformed shapes for different boundary condition

Node Label 1	RF.RF1 @Loc 1	RF.RF2 @Loc 1	U.U1 @Loc 1	U.U2 @Loc 1	S.S11 @Loc 1
1	0.	0.	66.2277E-06	-235.244E-06	690.466E+03
2	0.	0.	-34.3687E-06	-245.888E-06	-1.5625E+06
3	0.	9.5E+03	94.6110E-06	-7.50000E-33	-195.313E+03
4	0.	0.	47.3055E-06	-210.904E-06	-405.672E+03
5	0.	0.	47.3055E-06	-210.904E-06	694.444E+03
6	0.	0.	128.980E-06	-245.888E-06	-1.5625E+06
Minimum	-10.9139E-12	0	-34.3687E-06	-245.888E-06	-1.5625E+06
At Node	8	7	2	7	7
Maximum	0.	9.5E+03	128.980E-06	-7.50000E-33	694.444E+03
At Node	7	8	7	8	5
Total	-10.9139E-12	19.E+03	378.444E-06	-1.38407E-03	-1.84592E+06

Table 1: Data of truss figure 1 for different boundary condition

Table 1 shows all the output data form the ABAQUS for truss element for different boundary condition that is one corner is fixed support and another is roller support, these are applied on node 3 and 8. The force applied on node 3 and 8 is 2000N and node 2, 4, and 7 is 5000N.

RF (RF1, RF2) = Reaction forces at point 1 and 2, U (U1, U2) = Displacement, S11 = stress, E11 = strain

3.2. Truss Element: Both Roller Support

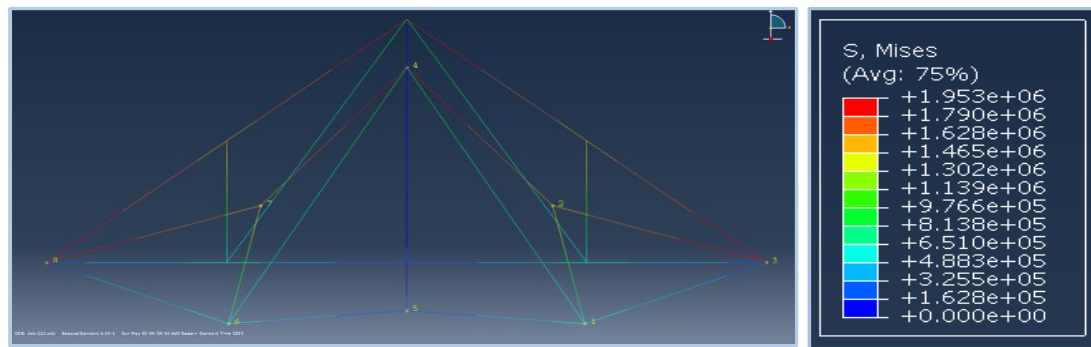


Figure 4: Deformed and undeformed shapes for same boundary condition

Node Label 1	RF.RF1 @Loc 1	RF.RF2 @Loc 1	U.U1 @Loc 1	U.U2 @Loc 1	S.S11 @Loc 1
1	0.	0.	-4.73055E-06	-187.938E-06	39.4239E+03
2	0.	0.	-93.5006E-06	-198.582E-06	-1.5625E+06
3	-8.33333E+03	9.5E+03	8.33333E-33	-7.50000E-33	-846.354E+03
4	0.	0.	0.	-147.830E-06	-405.672E+03
5	0.	0.	0.	-147.830E-06	-173.611E+03
6	0.	0.	4.73055E-06	-187.938E-06	39.4239E+03
7	0.	0.	93.5006E-06	-198.582E-06	-1.5625E+06
8	8.33333E+03	9.5E+03	-8.33333E-33	-7.50000E-33	-846.354E+03
Minimum	-8.33333E+03	0.	-93.5006E-06	-198.582E-06	-1.5625E+06
At Node	3	7.	2	7	7
Maximum	8.33333E+03	9.5E+03	93.5006E-06	-7.50000E-33	39.4239E+03
At Node	8.	8	7	8	6
Total	0.	19.E+03	-8.33333E-33	-1.06870E-03	-5.31814E+06

Table 2: Data of truss figure 2 for same boundary condition

Table 2 shows all the output data form the ABAQUS for truss element for same boundary condition that is both are roller support; these are applied on node 3 and 8. The force applied on node 3 and 8 is 2000N and node 2, 4, and 7 is 5000N.

RF (RF1, RF2) = Reaction forces at point 1 and 2, U (U1, U2) = Displacement, S11 = stress, E11 = strain.

3.3. Beam Element: Fixed and Roller Support

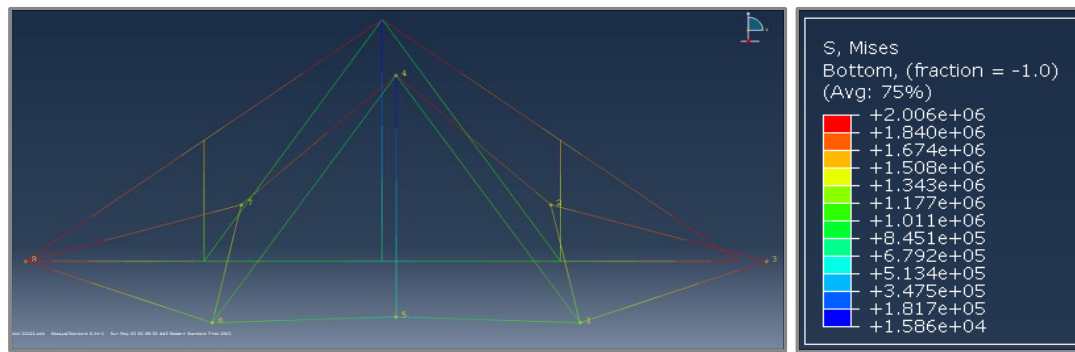


Figure 5: Deformed and undeformed shapes for different boundary condition

Node Label 1	RF.RF1@Loc 1	RF.RF2@Loc 1	U.U1 @Loc 1	U.U2 @Loc 1	S.S11 @Loc 5	S.S11 @Loc 6
1	0.	0.	66.2991E-06	-232.436E-06	781.625E+03	589.061E+03
2	0.	0.	-32.3841E-06	-242.841E-06	- 1.63731E+06	-1.45363E+06
3	0.	9.5E+03	94.5461E-06	-7.50000E-33	- 173.912E+03	-215.924E+03
4	0.	0.	47.2730E-06	-209.588E-06	- 438.100E+03	-375.591E+03
5	0.	0.	47.2730E-06	-210.020E-06	772.322E+03	634.761E+03
6	0.	0.	28.2470E-06	-232.436E-06	756.678E+03	614.008E+03
Minimum	0.	0.	-32.3841E-06	-242.841E-06	- 1.63731E+06	-1.45852E+06
At Node	7	7	2	7	2	7
Maximum	1.81899E-12	9.5E+03	126.930E-06	-7.50000E-33	781.625E+03	634.761E+03
At Node	8	8	7	8	1	5
Total	1.81899E-12	19.E+03	378.184E-06	-1.37016E-03	- 1.74503E+06	-1.88176E+06

Table 3: Data of beam for different boundary condition

Table 3 shows all the output data form the ABAQUS for beam element for different boundary condition that is one corner is fixed support and another is roller support, these are applied on node 3 and 8. The force applied on node 3 and 8 is 2000N and node 2, 4, and 7 is 5000N.

RF (RF1, RF2) = Reaction forces at point 1 and 2, U (U1, U2) = Displacement, S11 = stress, E11 = strain

3.4. Beam Element: Both Roller Support

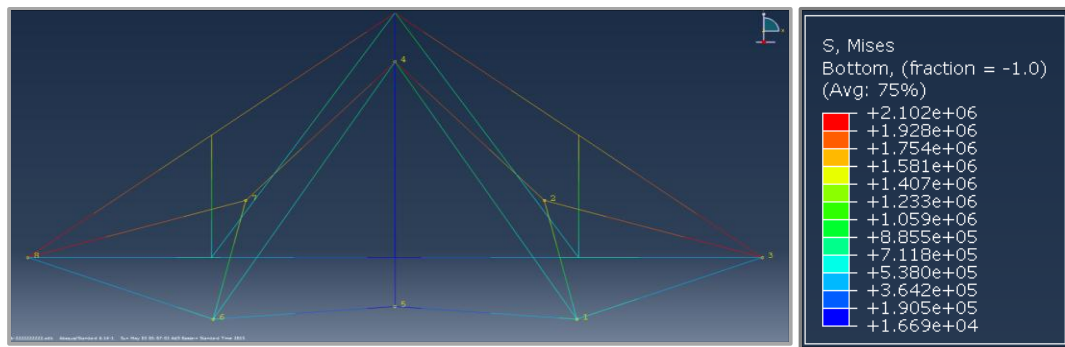


Figure 6: Beam Deformed and undeformed shapes for same boundary condition

Node Label 1	RF.RF1 @Loc 1	RF.RF2 @Loc 1	U.U1 @Loc 1	U.U2 @Loc 1	S.S11 @Loc 5	S.S11 @Loc 6
1	0.	0.	-4.61696E-06	-185.154E-06	111.000E+03	- 41.5951E+03
2	0.	0.	-91.4189E-06	-195.545E-06	-1.61841E+06	- 1.47246E+06
3	- 8.33391E+03	9.5E+03	8.33391E-33	-7.50000E-33	-828.393E+03	- 863.658E+03
4	0.	0.	94.0360E-21	-146.593E-06	-430.337E+03	- 383.643E+03
5	0.	0.	-15.0942E-21	-147.048E-06	-112.803E+03	- 214.957E+03
6	0.	0.	4.61696E-06	-185.154E-06	92.1891E+03	- 22.7837E+03
Minimu m	- 8.33391E+03	0.	-91.4189E-06	-195.545E-06	-1.61841E+06	- 1.47571E+06
At Node	3	7	2	7	2	7
Maximu m	8.33391E+03	9.5E+03	91.4189E-06	-7.50000E-33	111.000E+03	- 22.7837E+03
At Node	8	8	7	8	1	6
Total	0.	19.E+03	81.3152E-21	-1.05504E-03	-5.23030E+06	- 5.33846E+06

Table 4: Data of beam for same boundary condition

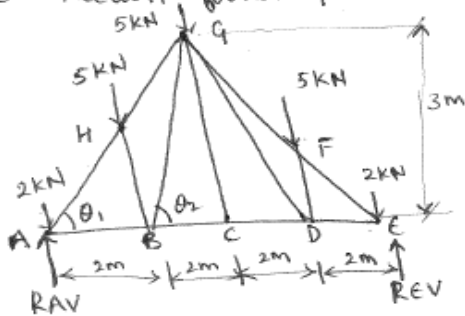
Table 4 shows all the output data form the ABAQUS for truss element for same boundary condition that is both are roller support; these are applied on node 3 and 8. The force applied on node 3 and 8 is 2000N and node 2, 4, and 7 is 5000N.

RF (RF1, RF2) = Reaction forces at point 1 and 2
U (U1, U2) = Displacement, S11 = stress, E11 = strain

4. THEORETICAL CALCULATION

Calculation:-

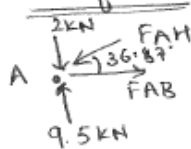
① Reaction forces & Member forces :-



Because of symmetry $RAV = REV = \frac{\text{Total load}}{2} = \frac{2+5+5+5+2}{2} = 9.5 \text{ kN}$

From triangle AGC $\tan \theta_1 = \left(\frac{3}{4}\right) \therefore \theta_1 = \tan^{-1}\left(\frac{3}{4}\right) \therefore \theta_1 = 36.87^\circ$

FBD of Joint A



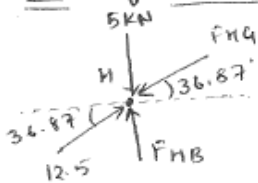
$$\begin{aligned} \sum F_y &= 0 \\ -2 + 9.5 - FAH \sin 36.87^\circ &= 0 \\ FAH &= 7.5 / \sin 36.87^\circ \therefore FAH = 12.5 \text{ kN (compression)} \end{aligned}$$

$$\begin{aligned} \sum F_x &= 0 \\ -12.5 \cos 36.87^\circ + FAB &= 0 \\ FAB &= 12.5 \cos 36.87^\circ \\ FAB &= 10 \text{ kN (Tension)} \end{aligned}$$

\therefore Similarly

$$\begin{aligned} F_{EF} &= 12.5 \text{ kN (compression)} \\ F_{ED} &= 10 \text{ kN (Tension)} \end{aligned}$$

FBD of Joint H



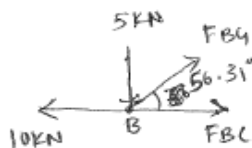
$$\begin{aligned} \sum F_x &= 0 \\ 12.5 \cos 36.87^\circ - FHG \cos 36.87^\circ &= 0 \\ FHG &= 12.5 \text{ kN (compression)} \end{aligned}$$

$$\begin{aligned} \sum F_y &= 0 \\ -5 + 12.5 \sin 36.87^\circ - 12.5 \sin 36.87^\circ + F_{HB} &= 0 \\ F_{HB} &= 5 \text{ kN (compression)} \end{aligned}$$

\therefore Similarly

$$\begin{aligned} F_{FG} &= 12.5 \text{ kN (compression)} \\ F_{FD} &= 5 \text{ kN (compression)} \end{aligned}$$

FBD of Joint B

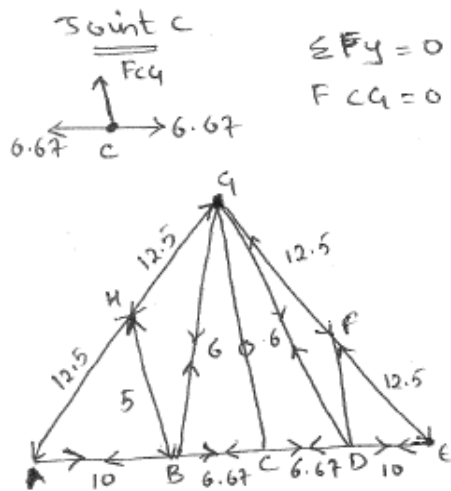


In triangle BCG $\tan \theta_2 = \frac{3}{2} \therefore \theta_2 = \tan^{-1}[1.5] \therefore \theta_2 = 56.31^\circ$

$$\begin{aligned} \sum F_y &= 0 \\ -5 + FBG \sin 56.31^\circ &= 0, \therefore FBG = 6 \text{ kN (Tension)} \end{aligned}$$

$$\begin{aligned} \sum F_x &= 0 \\ -10 + 6 \cos 56.31^\circ + F_{BC} &= 0, \therefore F_{BC} = 6.67 \text{ kN (Tension)} \end{aligned} \quad \textcircled{1}$$

similarly $F_{GD} = 6 \text{ kN (Tension)}$
 $F_{CD} = 6.67 \text{ kN (Tension)}$



Member	Length	Force	Type
AH, FE	2.5	12.5	Compression
HG, FG	2.5	12.5	Compression
HB, FD	1.5	5	Compression
BG, DG	3.61	6	Tension
AB, DE	2	10	Tension
BC, CD	2	6.67	Tension
CG	3	0	—

In triangle AHB
 $\cos \theta_1 = \cos 36.87^\circ = \frac{AH}{AB}$
 $AB = \frac{2}{\cos 36.87^\circ} = 2.5 \text{ m}$

In triangle AGC
 $\cos \theta_1 = \cos 36.87^\circ = \frac{AG}{AC}$
 $AC = \frac{2}{\cos 36.87^\circ} = 5 \text{ m}$
 $\therefore HG = AC - AH = 5 - 2.5 = 2.5 \text{ m}$

In triangle AHB
 $\sin \theta_1 = \sin 36.87^\circ = \frac{HB}{AB}$
 $HB = 2.5 \times \sin 36.87^\circ = 1.5 \text{ m}$

In triangle BGC
 $\sin \theta_2 = \sin 56.31^\circ = \frac{BG}{BC}$
 $BC = \frac{3}{\sin 56.31^\circ} = 3.61 \text{ m}$

5. VALIDATION

MEMBER	FORCE (N)	TYPE	STRESS (THEORETICALLY)	STRESS (ABAQUS)
AH	12.5	Compression	1953.125	-1.95E+06
FG	12.5	Compression	1953.125	-1.95E+06
HG	12.5	Compression	1953.125	-1.95E+06
FE	12.5	Compression	1953.125	-1.95E+06
HB	5	Compression	781.25	-7.81E+05
FD	5	Compression	781.25	-7.81E+05
BG	6	Tension	937.5	9.39E+05
DG	6	Tension	937.5	9.39E+05
AB	10	Tension	1562.5	1.56E+06
DE	10	Tension	1562.5	1.56E+06
BC	6.67	Tension	1042.1875	1.04E+06
CD	6.67	Tension	1042.1875	1.04E+06
CG	0		0	0.00E+00
			$F(x)=F/A \quad (A=0.0064)$	

Table 5: Comparison Between theoretical Calculation and Abaqus Results

The reaction forces for each member are calculated and forces are obtained. These forces are divided by the area that is 0.0064; hence we get stress theoretical values as shown in table. The truss element figure 1 is calculated from Abaqus and stress components i.e. s11. The theoretical values and Abaqus results are compared and both values are almost similar.

6. DISCUSSION

- The Purpose of this paper is to compare the results from ABAQUS and theoretical calculation. Though hand calculations are accurate but it is more complicated or nearly impossible to do it in some cases and time consuming and also increases computational cost. The use of ABAQUS software is much easier and reliable.
- The function of ABAQUS CAE is to produce approximate solution with satisfactory level of accuracy without providing unnecessary data.
- Simulation: The stresses, strains, reaction forces and even the deformed shape could be viewed using the ABAQUS simulation software. This comes handy in designing a new product as a lot of money can be saved by using this. When tested in the software if the design fails the company could go back and check or redo the design according to the safe parameters and requirements as per the software. The simulation software has many limitations. This analysis is generally used for modelling work and to construct contour plots of their results. It has also been observed that ABAQUS/CAE does not provide ideal representation of the analysis. However, it can be modified to view more accurate results, more easily to understand plots and tables.
- The truss and beam models are created in two- dimensional so the degree of freedom for these elements are two and three at each node.
- The truss element is pinned at the joint end point of the element, this act as a hinge and deforms at these points. In the case of beam, the structure is welded at the end points and when force is applied then they deforms at the nodes of the element. Seeding means the number of nodes within the element. In the beam element moment

force is induced. Truss will encounter an axial load in all members which leads to the same amount of force in entire member.

- On changing the mesh size from coarse to fine, a large region of small elements can be analysed critically. The accuracy of the results at the mesh corners is increased in fine refinement whereas in coarse, it is low. The computational efficiency is lower in coarse mesh and higher in fine mesh.
- Stress distributions for model Truss and Beam: It is observed that maximum stresses are applied at the nodes 1 and 6, where the maximum deformation is observed.

7. CONCLUSION

Since deriving stiffness matrix of a structure element using theoretical and mathematical equations for a complex geometry could be difficult and impossible, so we use the FEM analysis using ABAQUS to analyse it. In this report we use ABAQUS and FEM technique to solve the beam and truss structures. The steps for creating the element part were explained above. To have a better and clear understanding of the deformations and behaviours of the truss and beam element we calculated the values using hand calculations and then compared it with the results from ABAQUS. The mechanics of materials such as displacements, stresses and member forces are calculated by using ABAQUS/CAE. It was understood that the values from ABAQUS were more accurate than the hand calculated values. Higher accuracy can be achieved by meshing the element carefully and finely which can be time consuming and require much more processing. Finer mesh can be obtained but computational cost will increase and requires more time. In the case of beam element, it is difficult to calculate manually because the force is transmitted to each node member on deformation so ABAQUS is useful for complex structures. Thus FEM using ABAQUS helps us in understanding the deformations and strength of the different engineering materials used more accurately and easily.

REFERENCE

- [1] Takla, M 2015, "Introduction to the finite element method", Lecture notes at RMIT University, Melbourne, Australia.
- [2] Takla, M 2015, "Introduction to ABAQUS/CAE", Lecture notes at RMIT University, Melbourne, Australia.
- [3] Abaqus Version6.7 "ABAQUS Analysis User Manual Engineering forums"